

**STUDY OF VELOCITY PROFILE OF A CONVERGING COMPOUND  
CHANNEL USING ANSYS**

**A thesis submitted in the partial fulfilment of the requirement for the degree  
of bachelor of technology**

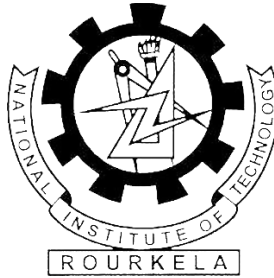
**In  
Civil Engineering**

**By  
RAHUL SAHOO  
ROLL NO. : 111CE0048**

**UNDER THE SUPERVISION OF  
Dr. K. K. KHATUA**



**DEPARTMENT OF CIVIL ENGINEERING  
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA  
ROURKELA, ODISHA, 769008 INDIA**



DEPARTMENT OF CIVIL ENGINEERING  
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

**CERTIFICATE**

This is to certify that the thesis entitled **Study of Velocity Profile of a Converging Compound Channel using ANSYS** submitted by **Mr. RAHUL SAHOO (111CE0048)** in the partial fulfilment of the requirement for the degree of Bachelor of Technology in Civil Engineering at National Institute of Technology Rourkela is an authentic work carried out by him under my supervision and guidance.

**Place: NIT Rourkela**

**Date:**

**Dr. K. K. Khatua**

**Department of Civil Engineering  
NIT Rourkela**

## **Acknowledgement**

I owe my profound respect and gratitude to my project guide, **Prof. K. K. Khatua**, Department of Civil Engineering, NIT Rourkela for his academic support and constant encouragement throughout the project. I could not be more thankful.

I would like to pay my gratitude to **Mrs. Bandita Naik**, research scholar, Department of Civil Engineering, NIT Rourkela for her constant guidance in the experimental and analytical field and help in carrying out this project.

**Rahul Sahoo**

**111CE0048**

## **ABSTRACT**

The three dimensional ANSYS Fluent modeling of a converging type non-prismatic compound open channel is worked out in this B Tech research project. The flow calculations are performed by using VOF (volume of fluid model). The channel dimensions are large enough to allow a steady flow, so transient flow type is considered as the potential type for precise mimicking of experimental conditions. The comprehensive validation of the ANSYS simulated results against the experimental data and checking whether the non-prismatic compound channels can be investigated using this software; is the major purpose of this research project. Analysis of open channel flow with converging floodplain is very much important because estimation of potential hazards becomes easier. The floodplain of any natural river is generally a densely populated area due to the obvious reasons of transportation and irrigation facilities. Hence the design of flood combating strategies are essential. Taking observations on a site of flooded river is also a risky and costly task so, after validation; use of ANSYS simulated models are a safe and cheaper option to analyze the river parameters. In this project, the results gave an understanding about the interaction of fluids flowing in main channel and floodplain.

### **Keywords**

Converging compound channel

Floodplain

Transient flow

Three dimensional ANSYS model

Flood combating strategy

# CONTENTS

| Topic                                       | Page |
|---|------|
| Certificate                                 | 1    |
| Acknowledgement                             | 2    |
| Abstract                                    | 3    |
| List of figures                             | 6    |
| List of Tables                              | 7    |
| Chapter 1: INTRODUCTION                     | 8    |
| Chapter 2: LITERATURE REVIEW                | 11   |
| Chapter 3: OBJECTIVE                        | 14   |
| Chapter 4: EXPERIMENTAL SETUP AND PROCEDURE | 15   |
| 4.1 Experimental layout                     | 15   |
| 4.2 Experimental setup                      | 15   |
| 4.3 Apparatus and Instrument used           | 17   |
| 4.4 Experimental Procedure                  | 18   |
| Chapter 5: NUMERICAL MODELING               | 19   |
| 5.1 Turbulence Model                        | 19   |
| 5.2 Governing Equation                      | 20   |

## CONTENTS

| Topic                                       | Page |
|---|------|
| Chapter 6:     NUMERICAL SIMULATION         | 21   |
| 6.1 Preprocessing                           | 21   |
| 6.2 Setup                                   | 22   |
| Chapter 7:     RESULT AND DISCUSSION        | 24   |
| 7.1 Comparison of Velocity Contour          | 24   |
| 7.2 Comparison of Vertical Velocity Profile | 25   |
| 7.3 Discussion                              | 27   |
| Chapter 8:     CONCLUSION                   | 29   |
| Chapter 9:     REFERENCES                   | 30   |

## LIST OF FIGURES

| Figure no. | Name of Figure  | Page no.  |
|------------|---|-----------|
| <b>1.</b>  | Natural open channel with floodplain  | <b>1</b>  |
| <b>2.</b>  | Locations of rain gauge stations  | <b>2</b>  |
| <b>3.</b>  | Above figure illustrates how number of (a) annual rain days and (b) annual rainy days decreasing over last 100 years. | <b>2</b>  |
| <b>4.</b>  | Layout of the experimental setup  | <b>8</b>  |
| <b>5.</b>  | cross sectional view of the channel   | <b>8</b>  |
| <b>6.</b>  | Experimental setup with Pitot tube and pointer gauge  | <b>10</b> |
| <b>7.</b>  | Experimental sections   | <b>11</b> |
| <b>8.</b>  | Geometry of the channel (b) Geometry after fluid fill.  | <b>14</b> |
| <b>9.</b>  | Meshing of the geometry   | <b>15</b> |
| <b>10.</b> | Comparison of section 1 velocity contour obtained by (a) Experiment and (b) Numerical analysis                        | <b>17</b> |
| <b>11.</b> | Comparison of section 2 velocity contour obtained by (a) Experiment and (b) Numerical analysis                        | <b>17</b> |
| <b>12.</b> | Comparison of section 3 velocity contour obtained by (a) Experiment and (b) Numerical analysis                        | <b>18</b> |
| <b>13.</b> | Comparison of vertical velocity profile obtained by Experiment and Numerical analysis at section 1                    | <b>18</b> |
| <b>14.</b> | Comparison of vertical velocity profile obtained by Experiment and Numerical analysis at section 2                    | <b>19</b> |
| <b>15.</b> | Comparison of vertical velocity profile obtained by Experiment and Numerical analysis at section 3                    | <b>19</b> |

## LIST OF TABLES

| Table no. | Name of the table   | Page no. |
|-----------|---|----------|
| 1.        | Details of Experimental parameters for Converging Compound<br>Channel | 9        |



## CHAPTER 1: INTRODUCTION

**Open channel flow** are very different from pipe flow in terms of complicity and randomness. Pipe flow are bounded flow generally driven by pressure whereas open channel flow are unbounded and flow from higher static head to the lower. The flow of liquid which is bounded at the bottom and possesses a free surface is called open channel flow. Such flow is driven by gravity and inertia of fluid mass. The free surface is technically an interface between the flowing fluid and the fluid above the surface that is atmosphere. The cross section of a Natural river generally consists of a main channel and a floodplain. A stage at which fluid flows in both main channel as well as floodplain, the flow is called compound open channel flow.

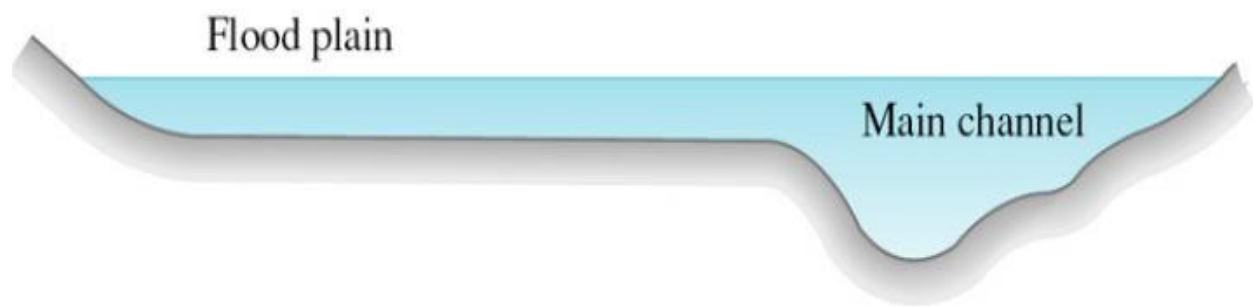


Fig. 1 Natural open channel with floodplain

The harsh meteorological conditions in India puts the country into a requirement of highly managed water resource system. The annual precipitation volume is increasing and the number of rainy days per annum are decreasing, which means storage of water resources and proper distribution is needed to avoid floods and ensure availability of water throughout the year. There are 2599 statins for recording precipitation data and the following figure shows the distribution of rain gauge stations throughout the country.

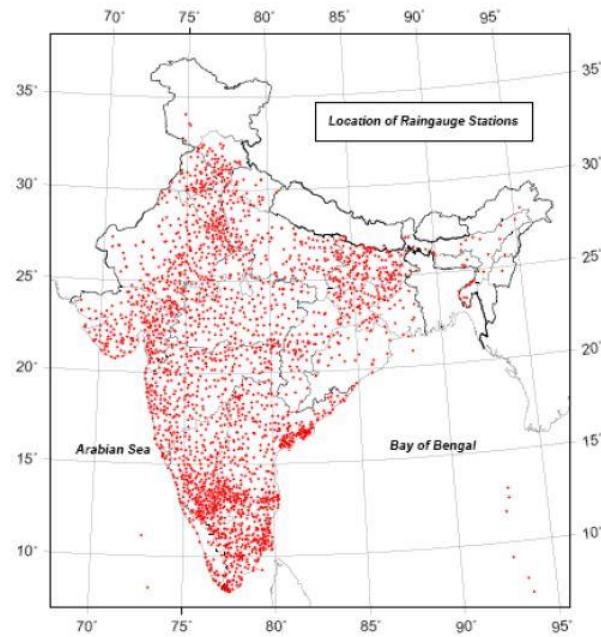


Fig 2. Locations of rain gauge stations

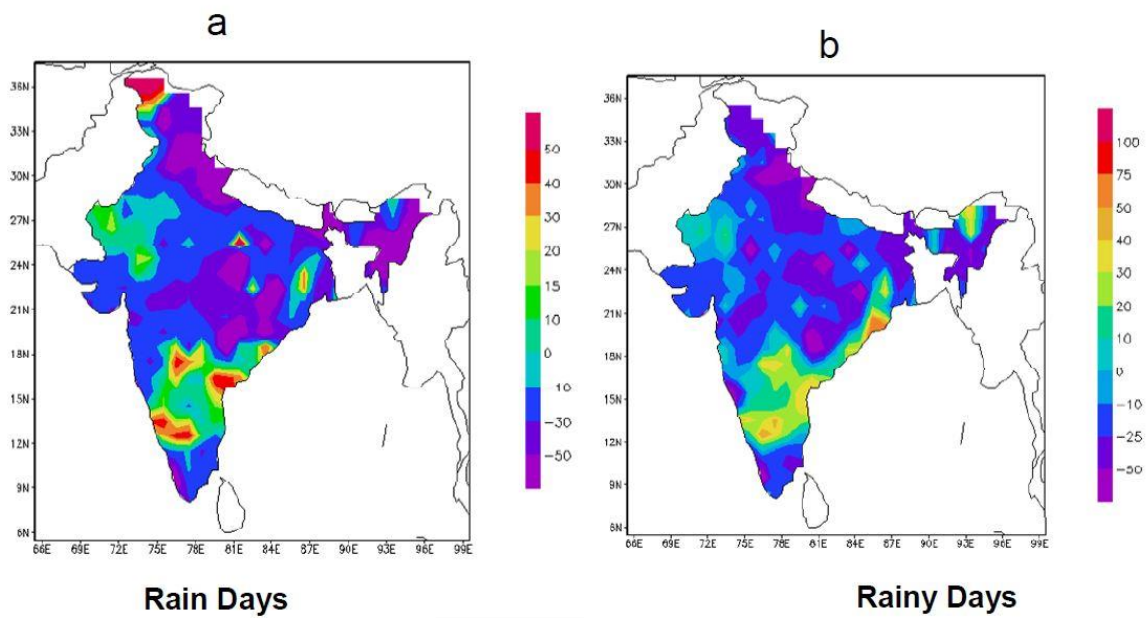


Fig 3. Above figure illustrates how number of (a) annual rain days and (b) annual rainy days decreasing over last 100 years.

Occurrence of flood depends upon intensity of precipitation. Increase in precipitation volume and decrease in rain days have contributed to increase in rainfall intensity. Flood occurrence also increases highly with increase maximum one day rainfall because it is the peak rainfall that causes the flood not the average rainfall. The following figure illustrates how maximum one day rainfall varied over the century.

The model used in this project is a **non-prismatic compound channel** which means the cross section of river is not uniform throughout the length of the channel. To be more specific, the floodplain of the model is converging type and finally merges to the main channel. The purpose of constructing such model is to mimic the natural river condition which passes through a populous area and its floodplain is reduced to null by human settlement and building construction. The destruction caused by the flood is generally due to release or loss of energy due to convergence in the floodplain. The amount of energy loss can be calculated by using the conservation of energy theorem.

The software used for numerical modeling in this case is computational fluid dynamics (CFD). There are many software packages on CFD like ANSYS and COMSOL. These software are based on finite element analysis. Simulation of CFD models are highly preferred because of its cost and time effectiveness.

## CHAPTER 2: LITERATURE REVIEW

The research work on compound channel with skewed floodplains were carried out by **James & Brown (1977)**. They worked on three different skew angle of  $7.2^\circ$ ,  $11.0^\circ$  and  $24.0^\circ$  and concluded that resistance to the flow increased with the skew angle and also explained the flow on the expanding floodplain accelerated while the flow on the converging floodplain decelerated.

**Johnson et.al. (1989)** investigated on attached flow in diffuser with small divergence angle. He concluded that for a diffuser half angle of less than  $5^\circ$ , the density current remains attached to both walls in a diffuser with horizontal bottom. He also concluded that if the densimetric Froude number is less than 2.0, the density current does not separate from the wall at diffuser half angle as large as  $40^\circ$ .

Further skewed channel experiments were done at the Flood Channel Facility (FCF) by researchers **Elliott & Sellin (1990)**, with three different skew angle of  $2.1^\circ$ ,  $5.1^\circ$  and  $9.2^\circ$ .

**Elliott (1990)** carried out further work at the Flood Channel Facility in the UK as part of the Series- A experiments on straight channels. He carried out detailed measurements of velocity, boundary shear stress and direction of flow.

The reduced conveyance of a skewed compound channel was confirmed by **Jasem (1990)** by compared it with a prismatic channel of similar cross-section.

**Ervine and Jasem (1995)** concluded that the velocity in the main channel is approximately constant or decreases slightly downstream. It causes a process of substitution occurs along the channel, due to the cross-over flow whereby the flow enters the main channel from the right floodplain must produce analogous removal of fluid from the main channel onto the left floodplain.

**Bousmar (2002) and Bousmar et al. (2004a)** analyzed the experiments on converging compound channels with symmetrically narrowing floodplains and explained about the

geometrical momentum transfer and the associated additional head loss due to symmetrically narrowing floodplains. They also estimated the additional head loss due to the mass transfer.

**Bousmar et al. (2004b)** also executed an additional investigations by using digital imaging to record surface velocities and horizontal turbulent structures that generally develop in prismatic channels.

**Proust (2005) and Proust et al. (2006)** investigated the flow analysis of a non-prismatic compound channel with asymmetric geometry with rushed convergence. They also found that a larger mass transfer and total head loss occurs at higher convergence angle as  $22^\circ$ .

**Bahram Rezaei (2006)** analyzed the experimental results of non-prismatic compound channels with converging floodplains. Due to change in floodplain geometry they found that the flow interaction of main channel and flood plain increases which causes large exchanges of momentum.

**Chlebek (2009)** has carried out a new experimental work on skewed channel and produced much more detailed data sets than the previously existing ones.

**Rezaei and Knight (2009)** developed a method for compound channels with non-prismatic floodplains by modifying the SKM method named as Modified SKM. In this the convergence effects were accounted by substituting the energy line slope ( $S_e$ ) with the channel bed slope ( $S_{0x}$ ).

**J.Chlebek, Bousmar et.al. (2010)** have explained the comparison of overbank flow conditions in skewed and converging/diverging channels. They observed that head losses increased due to the mass and momentum transfer and increased velocity gradient increased due to the expand of floodplains. They also observed the differences in the flow forcing from one subsection to another, velocity and bed shear stress measurements and significant differences in the flow distribution between main channel and floodplains.

**Proust et al. (2010)** estimated the energy losses in straight, skewed, divergent, and convergent compound channels by using first law of thermodynamics. They also concluded that the slope of energy line equals the head loss gradient at the total cross-section, yet the gradient of head loss differs with slope of energy line in the main channel or the floodplain.

**Rezaei and Knight (2011)** investigated the discharge distributions along three non-prismatic compound channel configurations for different converging angles. They also found that the discharge evolution seems linear for lower water depths; whereas non-linear for higher water depths and in the second half of the converging length the mass transfer is higher than that in the first half of the converging reach i.e. velocity increases significantly in the second half of the converging length.

**Khazaei & M. Mohammadiun (2012)** investigated three-dimensional and two phase CFD model for flow distribution in an open channel. He carried out the finite volume method (FVM) with a dynamic Sub grid-scale for seven cases of different aspect ratios, different inclination angles or slopes and convergence divergence condition.

**Anthony G. Dixon (2012)** simulated Computational fluid dynamics (CFD) software with fluid flow interactions between phases and he analyzed and improved it. He also checked the pressure drop and flow regime of a multiphase flow using CFD model.

The numerical simulation of a dam-break flow was performed by **Larocque, Imran, Chaudhry (2013)**. They used LES and k-  $\epsilon$  turbulence models and VOF (volume of fluid) model for free surface tracking.

## **CHAPTER 3: OBJECTIVE**

The aim of this work is to study the distribution of velocity profile in a non-prismatic converging compound channel. The present study focuses on the following aspects:

1. To conduct experimentations on non-prismatic compound channels of a converging angle to analyze the nature of change of flow variables throughout the non-prismatic reaches.
2. To study, the use of computational fluid dynamics (CFD) to predict the flow characteristics of non-prismatic open channels.
3. To study the turbulent flow structures of a non-prismatic compound channel flow using Large Eddy Simulation turbulent method.
4. The purpose of this project is to choose a computational fluid dynamics (CFD) program that would be utilized to simulate with the experimental convergent channel and to produce results on velocity distribution in non-prismatic channel system.

## CHAPTER 4: EXPERIMENTAL SETUP AND PROCEDURE

### 4.1 Experimental layout:

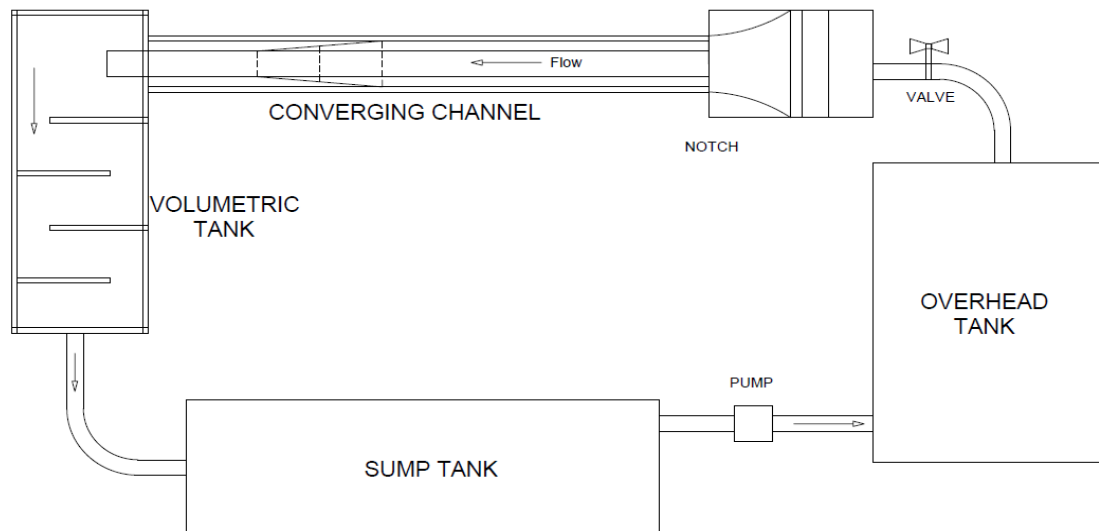


Fig. 4 Layout of the experimental setup

### 4.2 Experimental setup

#### 4.2.1 Flume:

Experiments were conducted in non-prismatic compound channels with varying cross section built inside a concrete flume measuring 15m×.90m×0.5m.

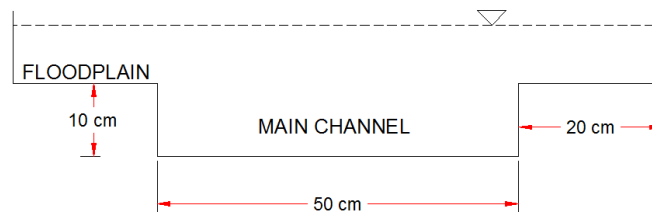


Fig. 5 cross sectional view of the channel



Table 1. Details of Experimental parameters for Converging Compound Channel

| Sl. No | Item Description  | Converging compound channel             |
|--------|---|---|
| 1      | Geometry of main channel  | Rectangular                             |
| 2      | Geometry of flood plain   | Converging                              |
| 3      | Main channel width( $b$ )   | 0.5m                                    |
| 4      | Bank full depth of main channel   | 0.1m                                    |
| 5      | Top width of compound channel ( $B_1$ ) before convergence                          | 0.9m                                    |
| 6      | Top width of compound channel ( $B_2$ ) after convergence                           | 0.5m                                    |
| 7      | Converging length of the channel  | 0.84m                                   |
| 8      | Slope of the channel  | 0.0011                                  |
| 9      | Width ratio( $\alpha$ ) = Ratio of top width ( $B$ ) to main channel width ( $b$ )  | $1 \leq \alpha \leq 1.8$                |
| 10     | Aspect Ratio ( $\delta$ )=Ratio of main channel width( $B$ ) to main channel height | 5                                       |
| 11     | Angle of convergence of flood plain ( $\emptyset$ )                                 | $12.38^\circ$                           |
| 12     | Flume size  | 15 m $\times$ .9 m $\times$ 0.5 m       |
| 13     | Position of experimental section 1  | At the start point of converging part   |
| 15     | Position of experimental section 3  | 0.42m forward from start of convergence |
| 17     | Position of experimental section 5  | At the end point of converging part     |

#### 4.2.2 Volumetric tank

Volumetric tank stores the fluid coming from the outlet of the flume for a temporary period of time. This tank helps us to calculate the actual discharged volume of water and thus discharge ( $Q$ ). This tank can be characterized by the partition walls which stabilizes the water surface so that the rise time of unit depth of water level can be recorded through a graduated cylinder connected externally to the tank.

Area of Volumetric Tank,  $A=20.928784 \text{ m}^2$

Let the time required for 1 cm rise of water be 't'.

Then the volume of water in t time  $V= A*0.1 \text{ m}^3$

And the discharge  $Q = V/t \text{ m}^3/\text{s}$

$$= (A*0.1)/t$$

#### **4.2.3 Sump tank and overhead tank**

Sump tank is the source of the water required in the experiment. Water is pumped out by an electric pump to the overhead tank. The pumps available in the lab are rated as 6hp and 2hp and the former was used in this experiment due to higher discharge requirements. Overhead tank was equipped with water depth sensor for safety purposes.

#### **4.3 Apparatus and instruments used**



Fig. 6 Experimental setup with Pitot tube and pointer gauge

Velocity measurement at the experimental sections are carried out using pitot tube which was connected to manometer with static and dynamic gauges. Pitot tube holder with 3 degree of freedom was installed so as to facilitate data collection at each grid point across the experimental section. The grid points at which point velocity was measured were spaced 5cm horizontally and 0.2 times the depth of flow vertically. Point gauges were installed at two locations of the channel to monitor the depth of flow. Tail gate was installed at the outlet of the channel to compensate the wavy nature of fluid flow.

#### 4.4 Experimental Procedure

In the setup of non-prismatic compound open channel with a converging angle of  $12.38^\circ$  (B Naik 2015) and non-prismatic length of 0.84 m, three experimental sections were chosen. Point velocities were taken at the grid points. The grid points in an experimental section are spaced horizontally by 5 cm and vertically by 0.2 multiplied by the depth of water at that point. Pitot tube was used to take these velocities which was connected to two piezometric heads through flexible pipes. One piezometer read static head and the other read total head, and their difference in reading is directly proportional to the point velocity. These piezometers are hung tilted to allow less error and increase accuracy in taking the readings. Pitot tube was shifted from one point to another by a roller and a hand operated rotating device. Steady uniform discharge was maintained the run of the experiment and several runs were conducted for overbank flow with relative depth varying between 0.15-0.51.

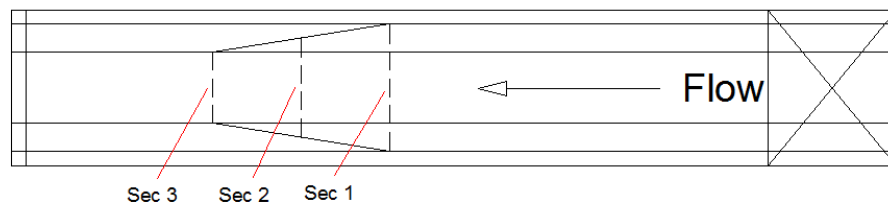


Fig. 7 Experimental sections

## **CHAPTER 5: NUMERICAL MODELING**

Computational fluid dynamics (CFD) is a computer programmed tool to analyze flow hydraulics using non-linear differential equations. Use of CFD modeling is gaining popularity in an exponential rate due to its easy adaptability and cost effectiveness because it avoids the costly experimental setups. It also involves zero human error. Hydraulic model facilitates mimicking various flood conditions and analyzing its various parameters without being at actual experimental sites. It reduces experimental risks by providing effortless simulations. The fundamental concept behind the CFD model is to analyze and solve a three dimensional flow by solving an number of governing equation by taking input as velocity, pressure, geometry and boundary conditions. In open channel flow, boundary conditions involves walls and bottoms providing slip-less shear resistance, the free surface as surface symmetry, inlet as velocity-inlet and outlet as pressure-outlet. In the present work, an effort has been made to investigate the velocity profiles for five different sections of a compound channel having converging flood plain by using a computational fluid dynamics (CFD) modeling tool, named as FLUENT. CFD uses FVM (finite volume method) for simulation and solution. LES (large eddy simulation) is generally used for channels with large dimensions.

### **5.1 Turbulence Models**

The naturally occurring compound open channel flows are generally turbulent in nature. This motion of fluid can be defined as an irregular flow which is often unpredictable. Unlike laminar flow the turbulent flow possesses a highly distorted. Followings are the CFD turbulence models:

1. Algebraic (zero-equation) model.
2.  $k$ - $\epsilon$ , RNG  $k$ - $\epsilon$  model.

3. Shear stress transport model.
4. K- $\omega$  model.
5. Reynolds stress transport model (second moment closure).
6. K- $\omega$  Reynolds stress.
7. Detached eddy simulation (DES) turbulence model.
8. SST scale adaptive simulation (SAS) turbulence model.
9. Smagorinsky large eddy simulation model (LES).
10. Scalable wall functions.
11. Automatic near-wall treatment including integration to the wall.
12. User-defined turbulent wall functions and heat transfer.

## 5.2 Governing Equation

The governing equation used in this simulation is K- $\omega$  turbulence model. This equation is based on energy conservation. This turbulence model is consist of two equations and is used as a closure to Reynolds-averaged Navier–Stokes equations (RANS equations). This model contains two partial differential equations one each variables K and  $\omega$ . K represents turbulence kinetic energy whereas  $\omega$  represents rate of dissipation of energy.

The partial differential equations are:

$$\begin{aligned}\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} &= P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_k \frac{\rho k}{\omega} \right) \frac{\partial k}{\partial x_j} \right], \quad \text{with } P = \tau_{ij} \frac{\partial u_i}{\partial x_j}, \\ \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} &= \frac{\gamma \omega}{k} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_\omega \frac{\rho k}{\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\rho \sigma_d}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}.\end{aligned}$$

Eddy viscosity  $\nu_T = K/\omega$ .

## CHAPTER 6: NUMERICAL SIMULATION

### 6.1 Preprocessing

Geometry of the given channel was created using design module (DM) tools in the CFD software. A frame of reference was chosen for the geometry for coordinate axes. Geometry was such that X-Z plane was parallel to the cross section of channel. Direction of flow was in the direction of Y-axis. Z-axis represented the vertical upward and X-axis represented the lateral direction of the geometry created. The angle of converging floodplain is  $12.38^\circ$ . After the geometry was created, fluid was filled using 'fill' tool and choosing the wetted area.

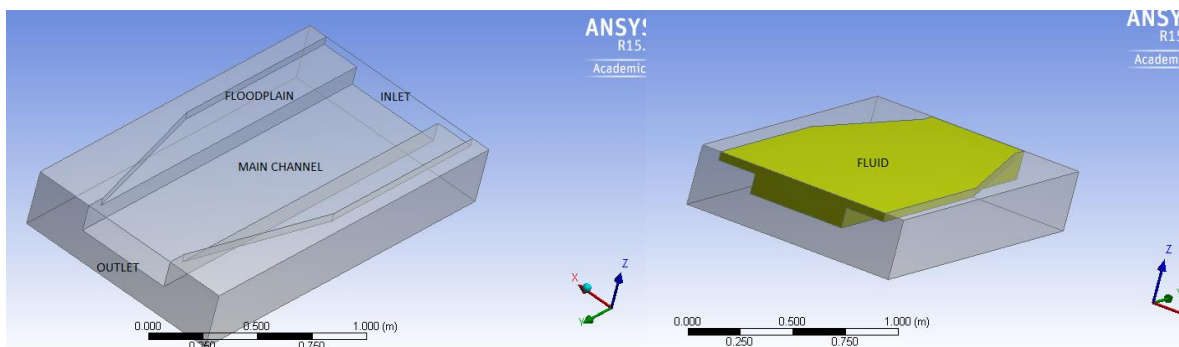


Fig. 8 (a) Geometry of the channel (b) Geometry after fluid fill.

Meshing was created with the help of Meshing tool in workbench. Sizing of the meshing was given as an input to the meshing tool to achieve a proper discretization of the model. High smoothing, slow transition and curvature normal angle =  $12^\circ$  are some preferable inputs to get a smooth meshing.

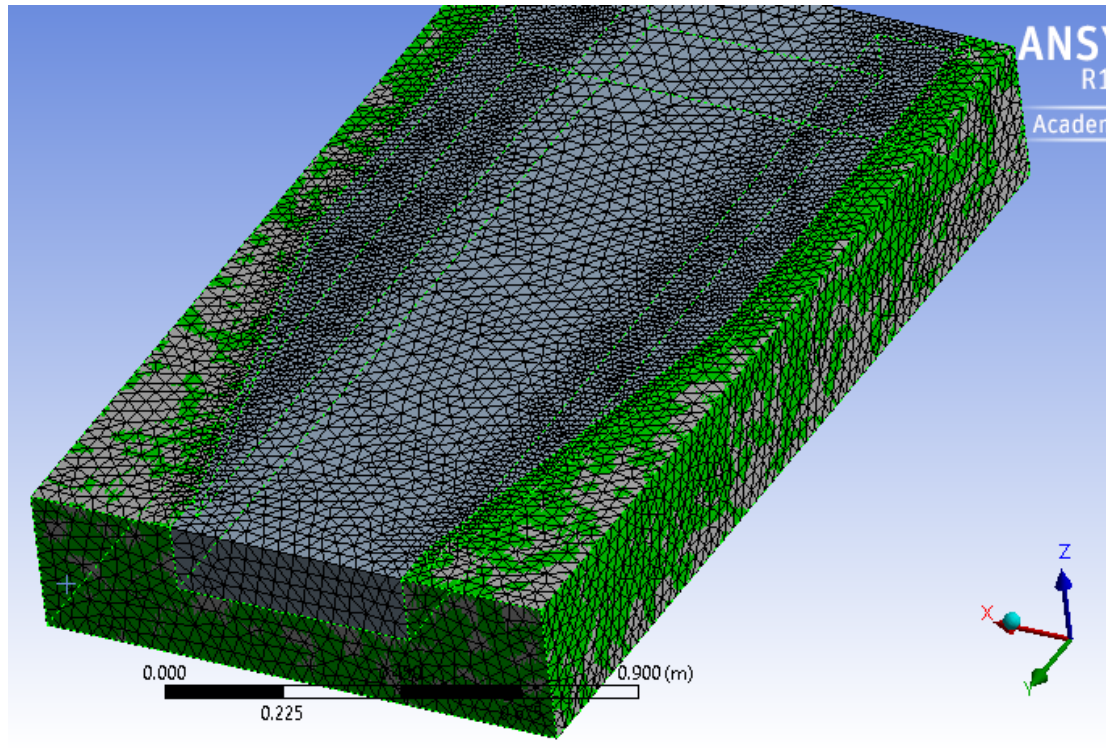


Fig. 9 Meshing of the geometry

## 6.2 Setup

After the meshing part is completed, various inputs are given to the Setup section. **VOF** (volume of fluid) model is the only model available for open channel flow simulation because this computational method deals with free surface flow. VOF is capable of calculating time dependent solutions.

Flow in an open channel is generally bound by channel from all directions except for the upward free surface. To achieve a free surface zero friction interface, a command called “surface\_symmetry” is given in named selection. Velocity inlet for inlet and pressure outlet for outlet is defined and the roughness coefficient is added to the walls for “no slip” condition.

Transient flow was chosen because the flow parameters were varying type with time in open channel experimentation. Gravity is check marked and the value for Z-axis is given as -9.81 because gravity acts downward opposite to the z-direction vector. As mentioned earlier, the turbulence model was chosen as k-omega model. Roughness coefficient generally varies from 0.5 to 1 where 0.5 is used for smooth surfaces. The roughness height for finished concrete varies from 0.3 mm to 3 mm. In this case it is taken as 1 mm. Among the solution methods PISO or SIMPLE is selected. PISO is generally a pressure-based segregated algorithm and it is recommended for transient flow conditions. It also allows a large time step for accurate calculation. Calculation is run from inlet after the initial values of pressure and velocity are given and y-velocity, volume fractions are patch. Time step size was set to 0.001 s and number of iteration given was 20 for better accuracy and convergence of the iteration.



## CHAPTER 7: RESULT AND DISCUSSION

Point velocities at grid points of the three different cross sections are collected and the velocity contour of each cross section is plotted using a software called Surfer. The CFD model of similar specifications as experimental channel was simulated and again the velocity contour is extracted. Velocity contours from both experimental and numerical analysis are compared. It was found out that the depth average velocities are close to each other with a standard deviation of 10-12%.

### 7.1 Comparison of Velocity contours

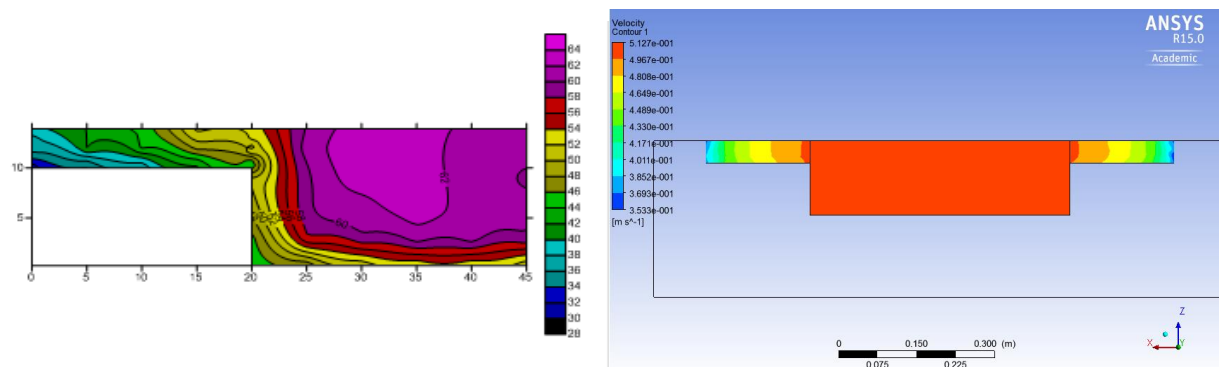


Fig. 10 Comparison of section 1 velocity contour obtained by (a) Experiment and (b) Numerical analysis

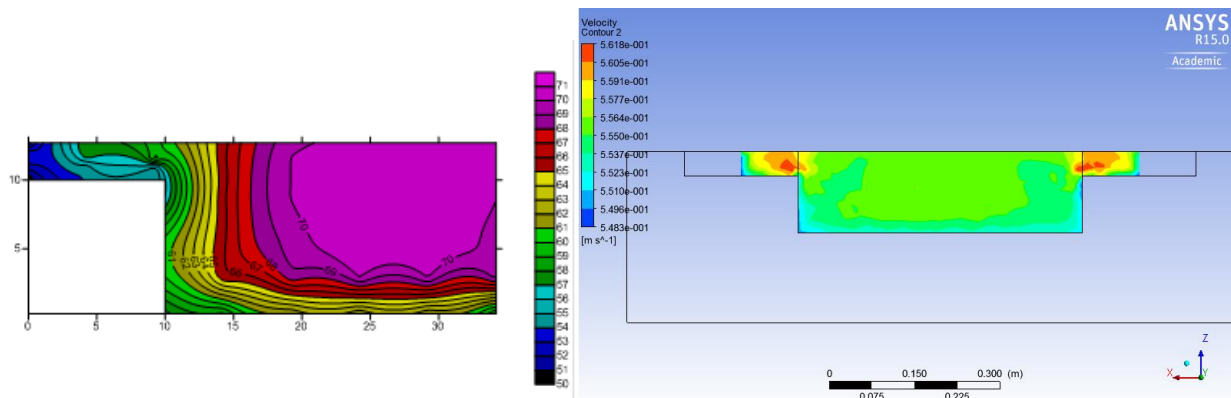


Fig. 11 Comparison of section 2 velocity contour obtained by (a) Experiment and (b) Numerical analysis

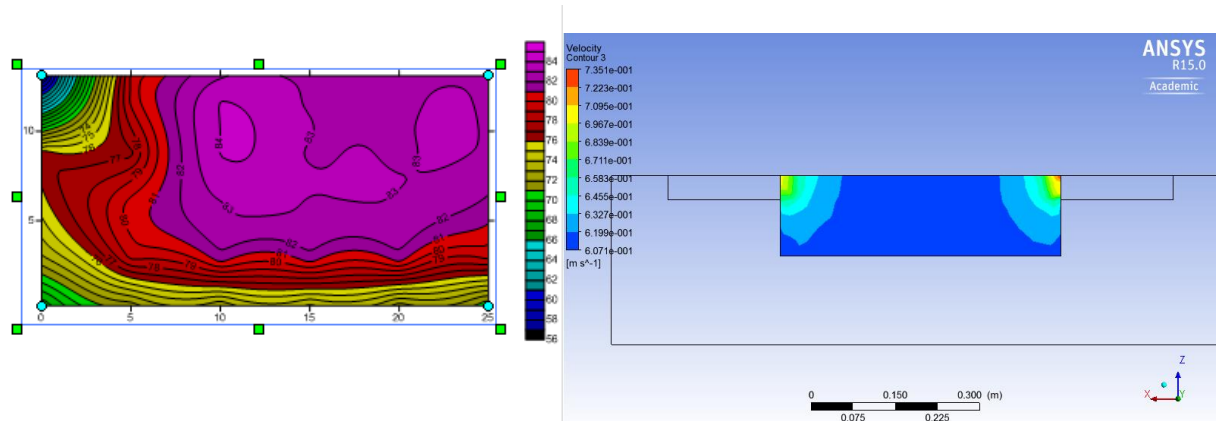


Fig. 12 Comparison of section 3 velocity contour obtained by (a) Experiment and  
(b) Numerical analysis

## 7.2 Comparison of vertical Velocity profile at the middle of the section

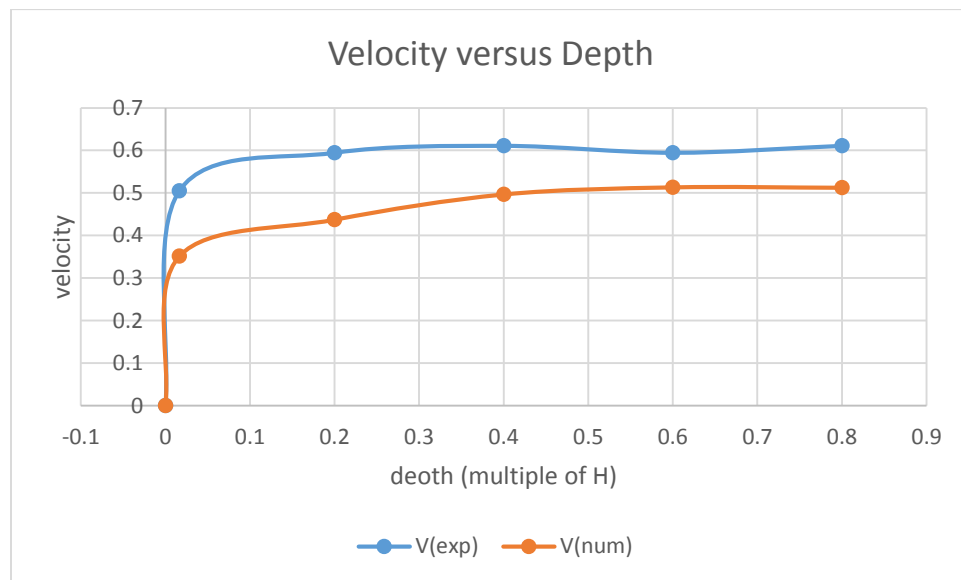


Fig. 13 Comparison of vertical velocity profile obtained by Experiment and  
Numerical analysis at section 1

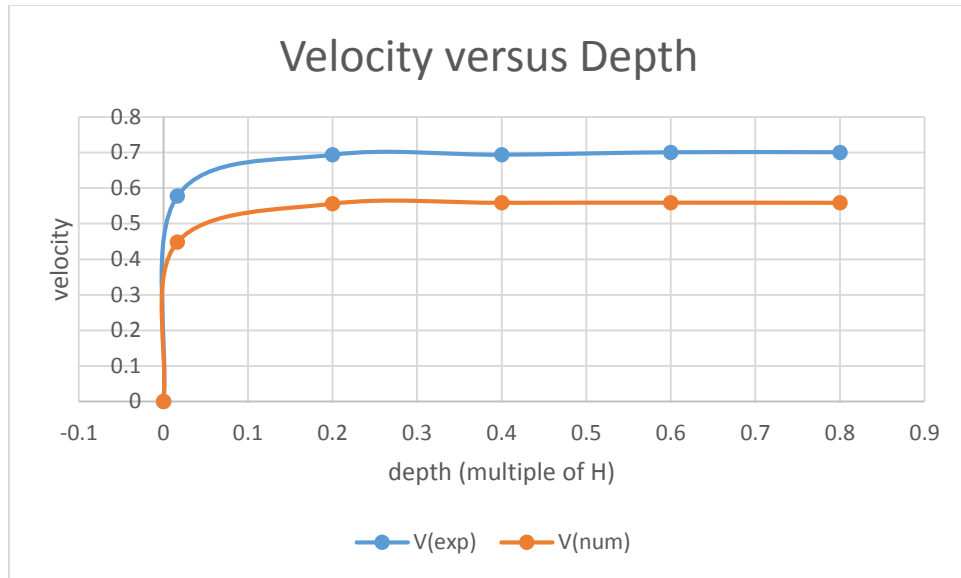


Fig. 14 Comparison of vertical velocity profile obtained by Experiment and Numerical analysis at section 2

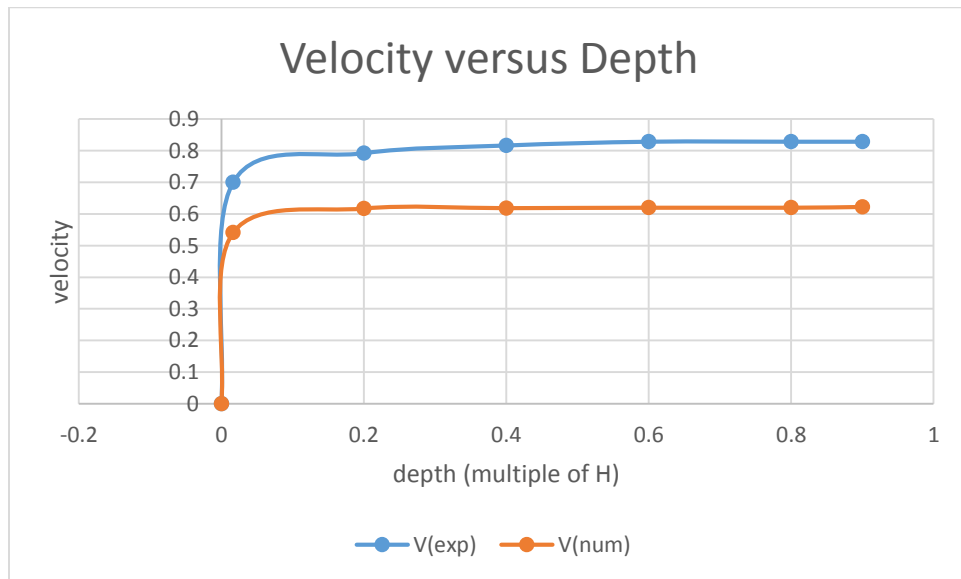


Fig. 15 Comparison of vertical velocity profile obtained by Experiment and Numerical analysis at section 3

### 7.3 Discussion

Point velocities at grid points of the three different cross sections are collected and the velocity contour of each cross section is plotted using a software called Surfer. The CFD model of similar specifications as experimental channel was simulated and again the velocity contour is extracted. Velocity contours from both experimental and numerical analysis are compared. It was found out that the point velocities are close to each other with a standard deviation of 10-12%. Figures 6,7 and 8 shows the longitudinal velocity contours at different sections along the convergence portion for 0.3 relative depth. As section 1 is located before the convergence, the velocity distribution is symmetric (similar to straight compound channels). Along the compression of the flood plain of the channel by the expansion of the major and minor secondary flows, it can be seen that at section 2 (middle of convergence) contours of the longitudinal velocity shift and the high velocity zone moves further towards the outer wall and the channel convergence. And at the section after the convergence (section 3), the high velocity zone is completely separated from the inner wall and transferred to the outer wall and the channel bed. Comparison of velocity contour between experimental data and numerical analysis of different sections are shown in Figures 6, 7 and 8 for  $Dr$  0.3.

Similarly the data obtained from Pitot tube measurements of the velocity were analyzed and velocity profiles were drawn. And then experimental analyzed results were compared and validated with numerical analyzed or CFD simulated results. Here the local velocities were also measured across the entire cross section, laterally every 50 mm and vertically at 0.2h, 0.4h, 0.6h, 0.8h, 0.9h level where “h” is the height of water for a particular section in the channel at three selected sections, at the beginning before convergence (section1) and end after convergence (section 3) and at the middle sections (section 2) inside the converging flume portion. The velocity

distributions at three experimental cross-sections for convergence angle of and  $Dr = 0.3$  are clearly shown that the CFD analyzed results are validated the experimental results.

## CHAPTER 8: CONCLUSION

In the present study, experiment has been conducted to study the velocity profile of a converging compound channel from section to section. The following conclusions are drawn:

- For a converging compound channel the vertical velocity profile was found to increase from section to section while moving from upstream to downstream.
- A numerical investigation has also been done using ANSYS to predict the vertical velocity profile. A good agreement of the result from ANSYS has been found with experimental results.
- The prediction from ANSYS was found to under-predict. The result can be improved by taking a higher inlet initial velocity.
- From the plot of velocities of both experimental and numerical analysis higher velocity occurs near the side of middle of main channel and minimum velocity was found in the floodplain.
- Similar trend found out in ANSYS but the ANSYS result was found to underestimate which may be due to lower initial velocity.
- The floodplain velocity increases from upstream to downstream in a converging channel due to significant reduction in floodplain area and momentum transfer.

## CHAPTER 9: REFERENCES

- Bousmar D, Wilkin N, Jacquemart J, Zech Y. Overbank Flow in Symmetrically Narrowing Floodplains, *Journal of Hydraulic Engineering*,130 (2004):pp. 305-312
- Bousmar D, Riviere N, Proust S, et al. Upstream discharge distribution in compound-channel flumes, *Journal of Hydraulic Engineering*,131 (2005):pp. 408-412
- Chlebek J. Modelling of simple prismatic channels with varying roughness using the SKM and a study of flows in smooth non-prismatic channels with skewed floodplains, University of Birmingham, 2009
- Chlebek J, Bousmar D, Knight DW, Sterling M. A comparison of overbank flow conditions in skewed and converging/diverging channels, *River flow international conference*, 2010:pp. 503-511
- Dixon AG, Walls G, Stanness H, Nijemeisland M, Stitt EH. Experimental validation of high Reynolds number CFD simulations of heat transfer in a pilot-scale fixed bed tube, *J. Chemical Engineering*, 200 (2012):pp. 344-356
- Elliott SCA, Sellin RHJ. SERC flood channel facility: skewed flow experiments, *Journal of Hydraulic Research*,28 (1990):pp. 197-214
- Ervine D.A, Jasem H.K. Observations on flows in skewed compound channels, *J. Wat. Marit. En., ICE*, 112 (1995): pp. 249-259.
- James, M. & Brown, B.J. (1977). “Geometric parameters that influence floodplain flow.” Report WES-RR-H, USACE, Vicksburg, USA, 77 (1977):pp. 1.
- Johnson TR, Ellis CR, Stefan HG. Negatively Buoyant Flow in Diverging Channel, IV: Entrainment and Dilution, *Journal of Hydraulic Engineering*,115 (1989):pp. 437-456

- Khazaee I, Mohammadiun M. Effect of flow field on open channel flow properties using numerical investigation and experimental comparison, International Journal of Energy & Environment, 2012:pp. 3
- Larocque LA, Imran J, Chaudhry MH. 3D numerical simulation of partial breach dam-break flow using the LES and k- $\epsilon$  turbulence models, Journal of Hydraulic Research, 51 (2013):pp. 145-157
- Proust S, Riviere N, Bousmar D, et al. Flow in Compound Channel with Abrupt Floodplain Contraction, Journal of Hydraulic Engineering, 132 (2006):pp. 958-970
- Rezaei B. Overbank flow in compound channels with prismatic and non-prismatic floodplains, University of Birmingham, 2006
- Rezaei B, Knight DW. Application of the Shiono and Knight Method in compound channels with non-prismatic floodplains, Journal of Hydraulic Research, 47 (2009):pp. 716-726
- Rezaei B, Knight DW. Overbank flow in compound channels with non-prismatic floodplains, Journal of Hydraulic Engineering, 137 (2010):pp. 815-824